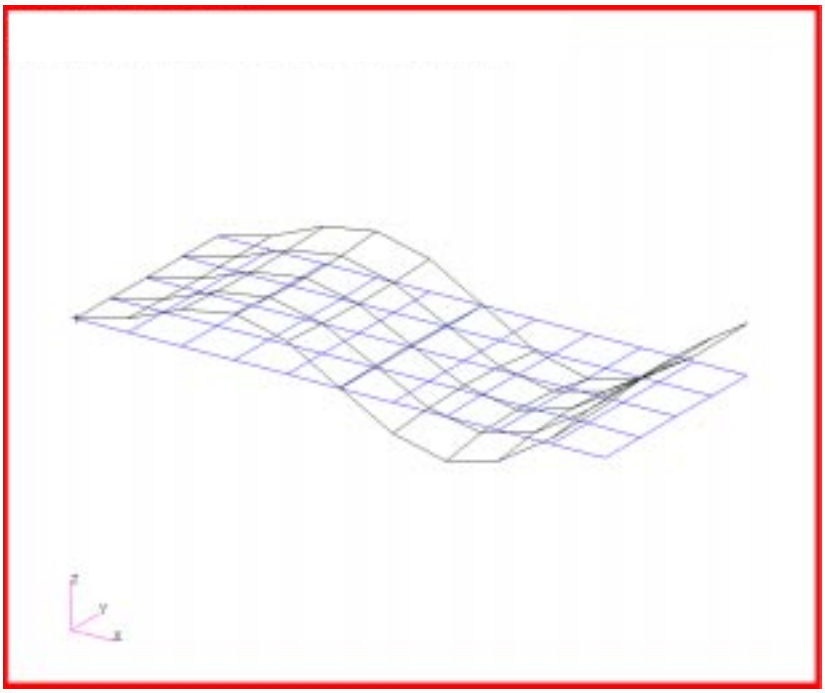
WORKSHOP PROBLEM 2

Modal Analysis of A Flat Plate using Static Reduction



Objectives

- Reduce the dynamic math model, created in Workshop 1, to one with fewer degrees of freedom.
- Apply the static reduction to the model.
- Submit the file for analysis in MSC/NASTRAN.
- Find the first five natural frequencies and mode shapes of the flat plate.

Model Description:

For this example, reduce the dynamic math model created in Workshop 1, using static reduction. Then find the first five natural frequencies and mode shapes using the Automatic Givens method. Use the points indicated in Figure 2.2 for the A-set.

Figure 2.1 - Grid Coordinates and Element Connectivities

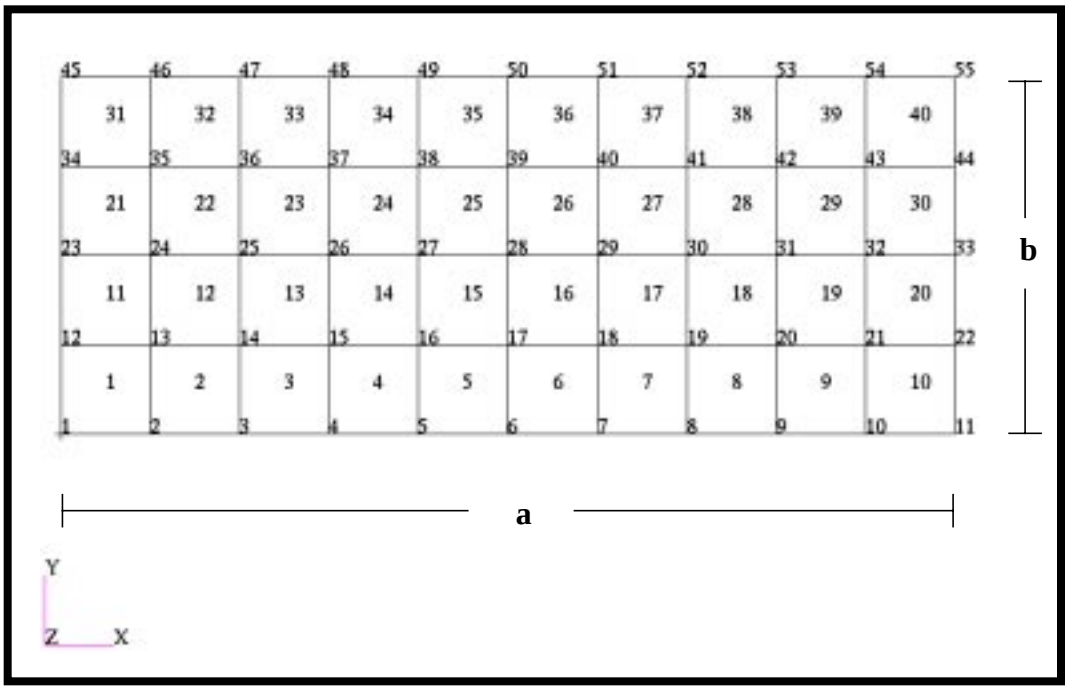


Figure 2.2 - Loads and Boundary Conditions

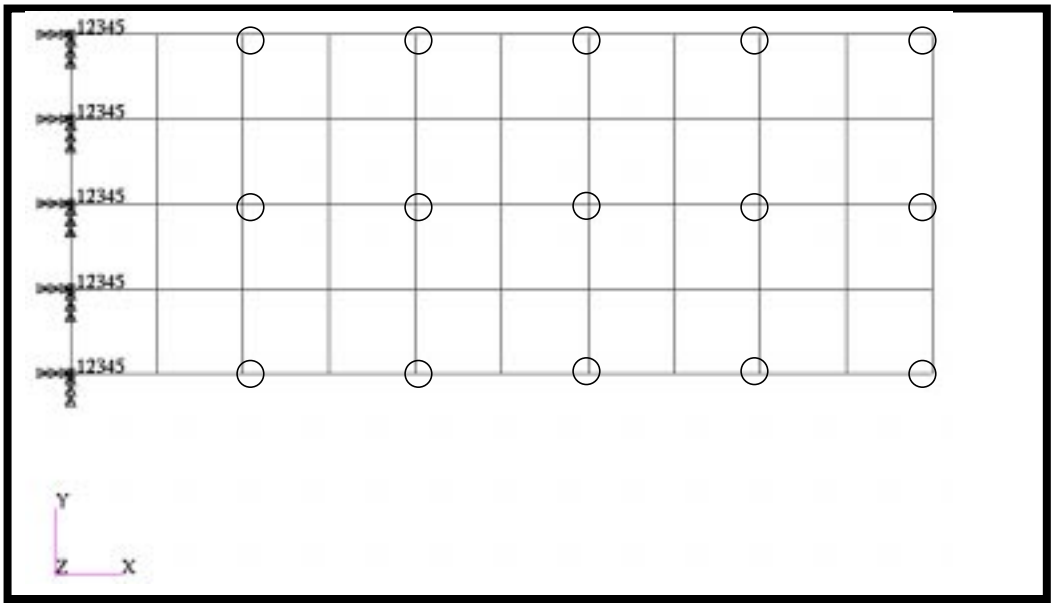


Table 2.1 - Properties

Length (a)	**5 in**
Height (b)	**2 in**
Thickness	**0.100 in**
Weight Density	**0.282 lbs/in^3**
Mass/Weight Factor	**2.59E-3 sec^2/in**
Youngs Modulus	**30.0E6 lbs/in^2**
Poisson's Ratio	**0.3**

Exercise Procedure:

1. Start up MSC/NASTRAN for Windows 3.0 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.

On the *Open Model File* form, select **New Model.**

Open Model File: **New Model**

2. Import **prob1.DAT.**

File/Import/Analysis Model...

● **Nastran** **MSC/Nastran**

OK

Change the directory to **C : \temp**.

File name: **prob1.DAT**

Open

When ask, “Ok, to Adjust all massess by PARAM, WTMASS factor of 0.00259?”, answer **No**. This information will be entered during analysis.

No

To reset the display of the model do the following:

View/Redraw

View/Autoscale

OK

3. Create the static reduction constraint.

Model/Constraint/Set...

ID: **2**

Title: **reduction**

OK

Now define the relevant constraint for the model.

Model/Constraint/Nodal...

Select the circled nodes by clicking on them, as shown in Figure 2.2 or manually input the selection as follows.

ID:	**3**	*to:*	**11**	*by:*	**2**	**More**
ID:	**25**	*to:*	**33**	*by:*	**2**	**More**
ID:	**47**	*to:*	**55**	*by:*	**2**	**More**

OK

On the *DOF* box, select these translational and rotational D.O.F.

☒ **TZ**

☒ **RX** ☒ **RY**

OK

Cancel

4. Create the input file for analysis.

File/Export/Analysis Model...

Type: **2..Normal Modes/Eigenvalue**

OK

Change the directory to **C:\temp**.

File name: **prob2**

Write

☒ **Run Analysis**

Advanced...

Modal Solution Method: ● **Lanczos**

Eigenvalues and Eigenvectors/ Number Desired: **5**

Mass: ● **Coupled**

OK

Problem ID: **Modal Analysis with Reduction**

OK

Under *Output Requests*, unselect all except:

☒ **Displacement**

Under *Analysis Case Request*, select the following:

☒ **Constraint (SPC) =** **1..NASTRAN SPC 1**

OK

Under *PARAM*, enter the following:

☒ **WTMASS** **.00259**

Under *Analysis Set*, enter the following:

☒ **ASET** **2..reduction**

OK

5. When asked if you wish to save the model, respond **Yes**.

Yes

File name: **prob2**

Save

When the MSC/NASTRAN manager is through running, MSC/NASTRAN will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time.

Continue

6. View the results of the analysis.

To get a better view of the deformation, you will want to rotate the model as follows:

View/Rotate...

Dimetric

OK

View/Select...

Deformed Style: ● **Deform**

Deformed and Contour Data...

Click on the Output Set listbox to get all modal frequencies. Answer the following questions:

What is the frequency for:

Mode 1 = _____
Mode 2 = _____
Mode 3 = _____
Mode 4 = _____
Mode 5 = _____

Compare these five results to the results of Workshop 1.

To view a mode shape, select one of the modes. View the results of the analysis.

OK

OK

When finished, exit MSC/NASTRAN for Windows.

File/Exit

This concludes this exercise.

Mode 1	**133.70 Hz**
Mode 2	**690.24 Hz**
Mode 3	**844.87 Hz**
Mode 4	**2225.95 Hz**
Mode 5	**2449.03 Hz**